All Sciences Proceedings
http://as-proceeding.com/

3rd International Conference on Innovative Academic Studies

September 26-28, 2023 : Konya, Turkey

ICIAS
https://www.icias.net/

Modal Analysis of Five-Storey Reinforced Concrete Structure Using MATLAB Partial Differential Equation Toolbox

Olgun KÖKSAL[1*], Zeki KARACA[2] and Erdem TÜRKELİ[3]

[1] *Lecturer Doctor, Kavak Vocational Junior College, Samsun University, Samsun, Turkey*
[2] *Prof. Dr., Civil Eng. Depart., Ondokuz Mayıs University, Samsun, Turkey*
[3] *Assoc. Prof. Dr., Ordu University Vocational School of Technical Sciences, Construction Depart., Ordu, Turkey*

*Corresponding Author: * olgun.koksal@samsun.edu.tr*

Abstract – Different methods such as the equivalent earthquake load method, response spectrum analysis, time domain analysis, and frequency domain analysis are used to determine the seismic behavior of reinforced concrete (RC) structures. Response spectrum analysis, one of these methods, is frequently preferred and used by civil engineers. Turkey Building Earthquake Regulations (entered into force on January 1, 2019) also includes information on the use of this cited method. It is stated that the response spectrum analysis of RC structures can be performed with horizontal and vertical elastic design spectra to be generated according to different earthquake ground motion levels and soil classes using the earthquake hazard map. In this study, a five-story RC building is produced. The produced RC building consists of structural elements such as columns, beams, slabs, shear walls, and a raft foundation. Modal analysis was performed by using MATLAB partial differential equation (PDE) toolbox based on the finite element method. The mode shapes and frequencies of the structure were determined by modal analysis. Then the obtained analysis results obtained from MATLAB PDE toolbox were compared with the ones obtained by using ANSYS software.

Keywords – Turkish Building Earthquake Code, Reinforced Concrete Structure, Finite Element Method, Modal Analysis

I. INTRODUCTION

Different methods such as equivalent earthquake load method, response spectrum analysis, time domain analysis, and frequency domain analysis are used in obtaining the earthquake behaviour of RC structures. Response spectrum analysis can be counted as one of the most preferred and utilized methods for the dynamic analysis of structures under earthquake effects [1-6]. The response spectrum is a graph of the maximum response (maximum displacement, velocity, acceleration, or any other quantity of interest) to a given load function for all possible single-degree-of-freedom systems. The abscess of the spectrum is the natural frequency (or period) of the system, and its ordinate is the maximum response [1]. In spectrum analysis, the amplitude of each mode motion is found from the spectrum curves. These amplitudes are then combined to calculate the critical motion of the structure. When this method is used, results are obtained more easily without solving the equations of motion numerically or implicitly [6]. Also, in order to use this method, free vibration analysis of the structures must be performed to obtain mode shapes and frequencies. In addition, free vibration analysis should also be performed for the damping matrix obtained depending on the mass and stiffness of the structure. Nowadays, there are many software for numerical modeling of structures [7]. However, most of these software are not open source, and for researchers working with open-source software provides many benefits. Software based on the Finite Element Method (FEM) [7] is preferred for solid modeling of structural systems. Mechanical properties, meshing, and initial and boundary conditions affect the analysis results while modeling the structures with FEM. In this study, the free vibration analysis of a RC five-storey structure is carried out with the FEM. The analyses were performed with both MATLAB PDE toolbox [8] and ANSYS [9] software and the results were compared with each other.

II. Materials and Method

A. *MATLAB Partial Differential Equation (PDE) Toolbox*

PDE Toolbox (Fig.1) provides functions for solving structural mechanics, heat transfer, and general PDEs using the FEM [8].

Features of this cited toolbox [8] ;

•It can be used to calculate deformations and stresses.

•For modeling the dynamics and vibration of the structure, the toolbox has a time-integrating solver directly.

•It can analyze the structural properties of a component by performing modal analysis to find natural frequencies and mode shapes.

•It can model conductive heat transfer problems to calculate heat distributions, heat flow and heat flow rates over surfaces.

•It can also solve standard problems such as diffusion, electrostatics, magnetostatics and special PDEs.

•Imports 2D and 3D geometries using mesh data.

•It can automatically create meshes with triangular and tetrahedral elements.

The PDE toolbox calculates equations in the form given below [8]

$$m\frac{\partial^2 u}{\partial t^2} + d\frac{\partial u}{\partial t} \cdot \nabla \cdot (c\nabla u) + au = f \qquad (1)$$

$$\nabla \cdot (c\nabla u) + au = \lambda du \qquad (2)$$

$$\nabla \cdot (c\nabla u) + au = \lambda^2 mu \qquad (3)$$

solves eigenvalue problems. When solving PDEs, there are two boundary choices for each edge or face. Dirichlet boundary conditions performs the solution equation at the edge or surface.

$$hu = r \qquad (4)$$

where, h and r are denoting 3D (x, y, z) space functions. Generalized Neumann boundary conditions performs the solution equation at the edge or surface.

$$\vec{n} \cdot (c\nabla u) + qu = g \qquad (5)$$

where, $\vec{n}$ is denoting the unit normal vector. q and g are functions defined in $\partial\Omega$ depending on (x, y, z) in 3D space.

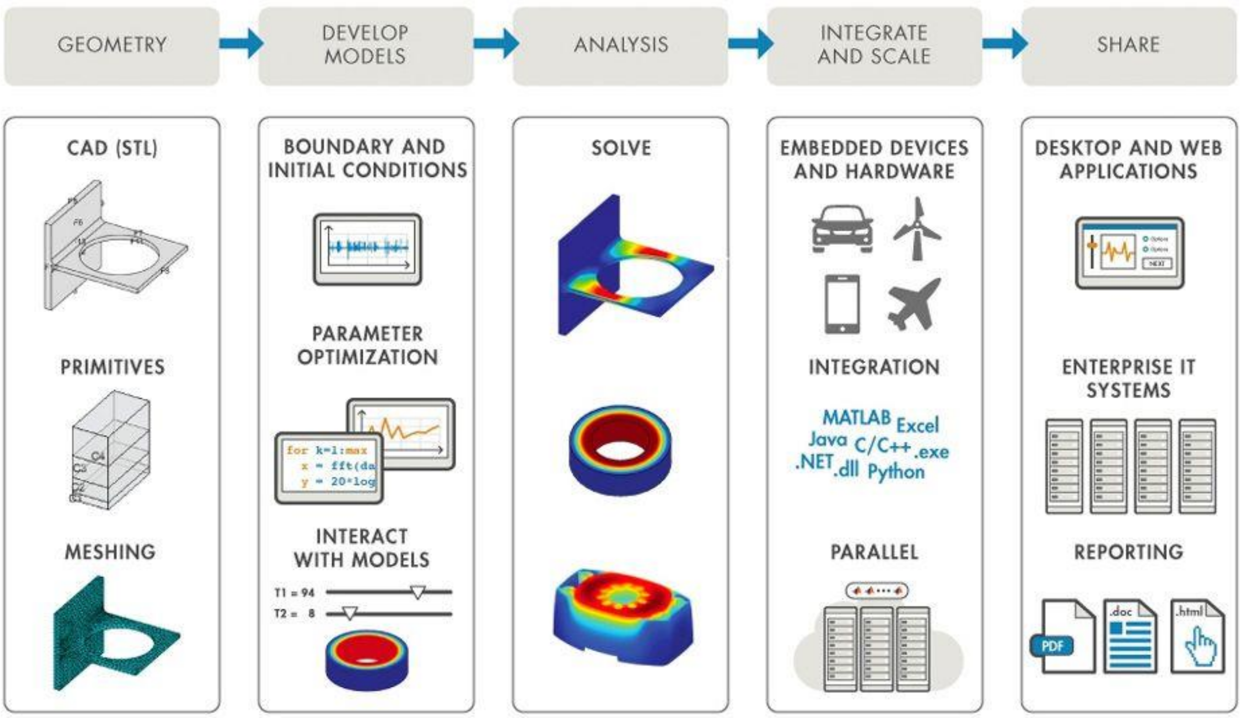
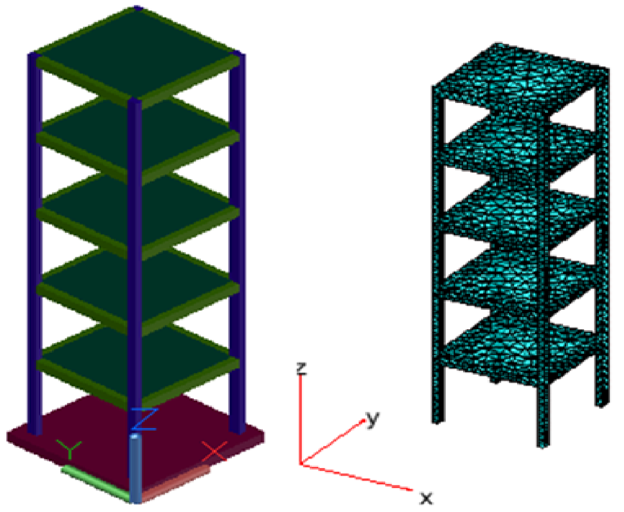


Fig. 1. The flowchart of PDE Toolbox [8]

B. Application

In this study, a five-storey RC structure is produced in before cited softwares and free vibration analyses are performed and compared with each other. The five-storey RC structure consists of columns, beams, slabs and a raft foundation. The geometrical dimensions and material properties of the structure are given in Table-1 and Table-2, respectively. C25 concrete class was assumed for the modeling. In the model, linear elastic material behavior was assumed and the stiffness degradation was neglected. The steel reinforcing bar was not modeled. The solid model and FEM of the structure is given in Fig.2. Linear tetrahedral solid element was used as finite element meshing. These elements have three degrees of freedom at each node. The finite element model includes 2584 nodes and 6375 solid finite elements in total. The structure is assumed to be fixed to the ground.



(a) (b)

Fig. 2 (a) Solid model and (b) FEM of RC structure

Table 1. Geometrical properties of RC structure

Column dimensions	30cmx45cm
Column height	4m
Beam dimensions	30cmx45cm
Slab thickness	15cm
Foundation thickness	50cm

Table 2. Material properties of the cited RC structure [10]

Material	Concrete
Class	C25
Modulus of Elasticity	30GPa
Poisson Ratio	0.20
Unit weight of concrete	24kN/m^3

Table 3. Comparison of modal analysis results

Mode number	**Modal frequency (MATLAB PDE Toolbox)**	**Modal frequency (ANSYS)**	**Difference (%)**	**<%6**
1	1.49	1.41	5.60	✓
2	1.80	1.86	3.33	✓
3	2.42	2.50	3.33	✓
4	4.57	4.51	1.30	✓
5	5.65	5.89	4.20	✓
6	7.43	7.84	5.50	✓

III. RESULTS

Free vibration analyses for the five-storey RC structure were performed with both MATLAB PDE toolbox and ANSYS software. The analysis results are shown in Fig.3 and Fig.4, respectively. As can be seen from Fig.3 and Fig.4, the first mode of the structure is displacement in x direction, the second mode is displacement in y direction and the third mode is the torsion mode. By examining Figs. 3 and 4, it can be clearly seen that according to the analysis performed in MATLAB PDE toolbox, the frequency of the first, second, third, fourth, fifth and the sixth mode are 1.4985Hz, 1.8027Hz, 2.421Hz, 4.5758Hz, 5.6562Hz and 7. 4344Hz, respectively. Also, the same examination (Figs. 3 and 4) of ANSYS reveals for the frequency of the first, second, third, fourth, fifth and the sixth mode are 1.4187Hz, 1.8677Hz, 2.501Hz, 4.51Hz, 5.8918Hz, 7.8424Hz, respectively.

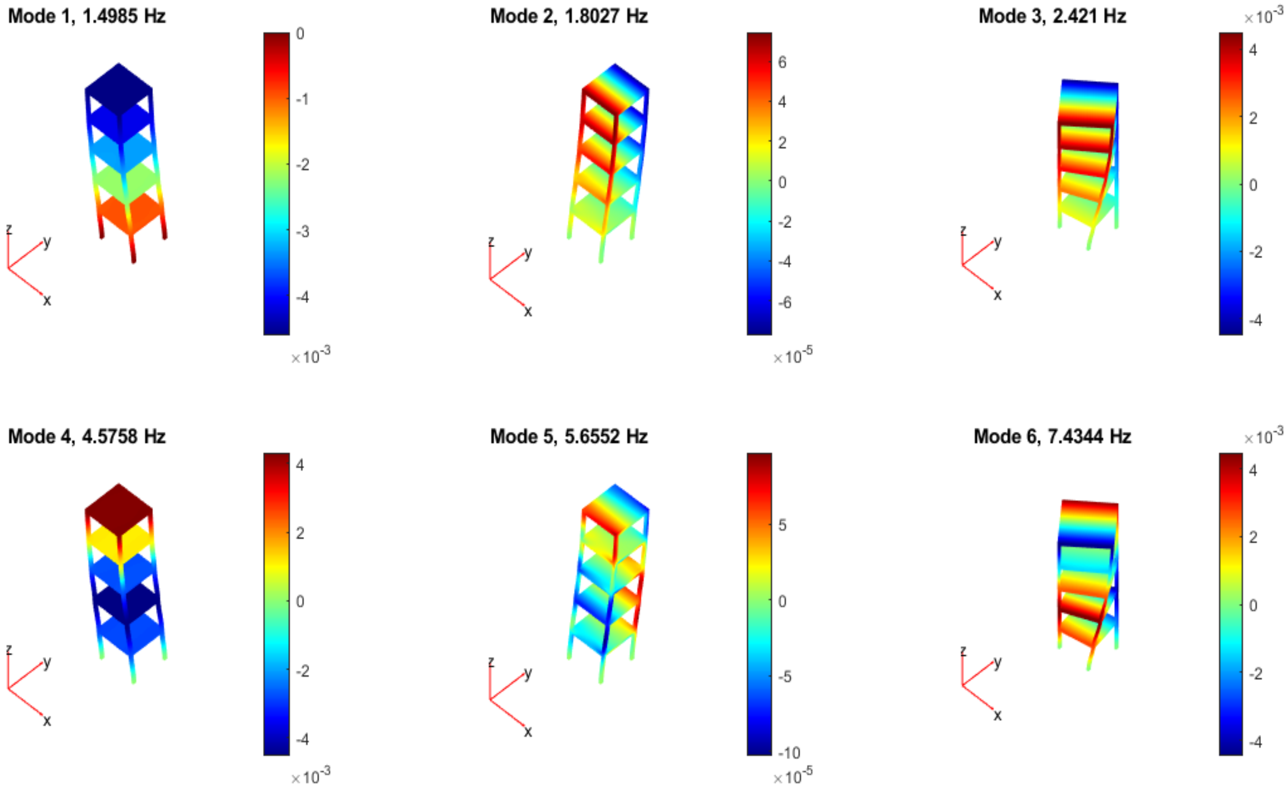


Fig. 3 Mode shapes and frequencies of RC structure (from MATLAB PDE Toolbox)

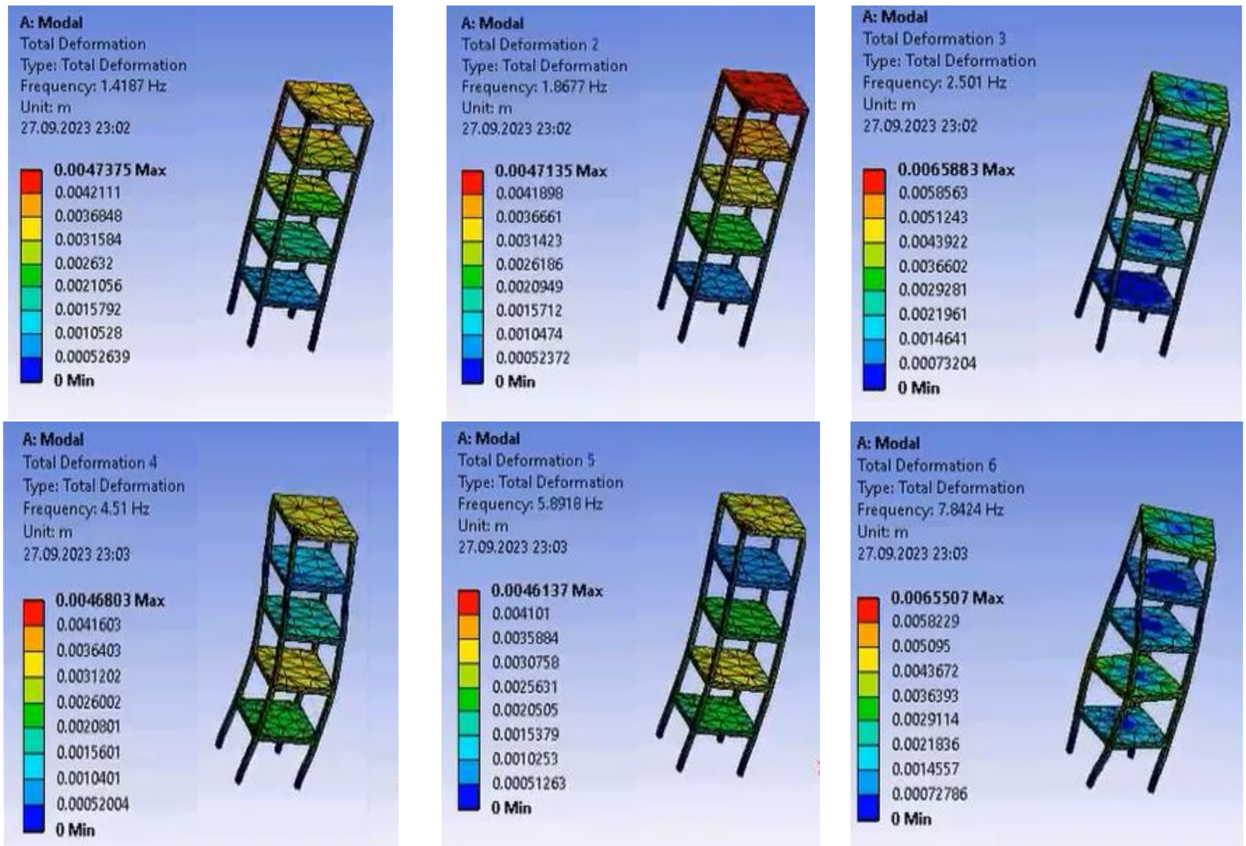


Fig. 4 Mode shapes and frequencies of RC structure (from ANSYS)

IV. DISCUSSION

Table 3 was prepared by examining the results given in Figs. 3 and 4 together. Table 3 shows the comparison of modal frequencies of the five-storey RC structure obtained from MATLAB PDE Toolbox and ANSYS. It is seen that the difference between the results of these cited softwares is less than 6%. This difference is thought to be due to reason that the number of finite elements and meshing properties are different.

V. CONCLUSION

In this study, the free vibration analysis of a five-story RC structure is carried out by FEM. The analyses were performed both with MATLAB PDE toolbox and ANSYS software. As a result of the analysis, mode shapes and frequencies of the cited structure were obtained. It is seen that the mode shapes are in the form of displacement in x direction in the first mode, displacement in y direction in the second mode and torsion in the third mode. This is consistent with the literature. Also, similar numerical and visual results were obtained in both software. The difference between the frequencies is less than 6% and this difference is due to number and properties of finite element meshing. According to these obtained results, it can be deducted that MATLAB PDE Toolbox can be utilized in structural analysis of building type of RC structures. However, the results obtained in this study belong to one specific building type of RC structure. Therefore, the number of analysis and building types should be increased in order to generalize the results.

REFERENCES

[1] M. Paz and W. Leigh, "Structural Dynamics," 2004, doi: 10.1007/978-1-4615-0481-8.

[2] G. P. Cimellaro and S. Marasco, *Introduction to dynamics of structures and earthquake engineering*. Springer, 2018.

[3] R. W. Clough and J. Penzien, "Dynamics of Structures. 2nd edition," ed: New York: McGraw-Hill Education, 1993.

[4] J. T. Katsikadelis, *Dynamic analysis of structures*. Academic press, 2020.

[5] "Turkish Building Earthquake Code-2018," ed.

[6] A. K. Chopra, *Dynamics of structures*. Pearson Education India, 2007.

[7] A. A. Kasımzade, *Finite Element Method Fundamentals and Applications in Structural Mechanics*. Nobel Academic Publishing, 2018, p. 848.

[8] MATLAB R2023a. (2023). https://www.mathworks.com/products/pde.html

[9] ANSYS (2014). Finite Element Analysis System, SAS IP, Inc., US

[10] A. Doğangün, "Calculation and Design of Reinforced Concrete Structures," 2020.